

Available online at www.sciencedirect.com**ScienceDirect**

Energy Procedia 75 (2015) 3174 – 3179

Energy
Procedia

The 7th International Conference on Applied Energy – ICAE2015

A numerical study on the shell-side turbulent heat transfer enhancement of shell-and-tube heat exchanger with trefoil-hole baffles

Guo-Yan Zhou*, Jingmei Xiao, Lingyun Zhu, Juntao Wang, Shan-Tung Tu

Key Laboratory of Pressure Systems and Safety(MOE), East China University of Science and Technology, Shanghai, 200237, China

Abstract

Shell-and-tube heat exchangers with trefoil-hole baffles are new type heat transfer devices and widely used in nuclear power system due to their special advantages, with the fluid flowing longitudinally on the shell side. However, very few related academic literature are available. In order to obtain an understanding of the underlying mechanism of shell-side thermal augmentation, a CFD model including inlet and outlet nozzles is proposed in the present study. Based on the RNG k- ϵ model, numerical investigations on shell-side fluid flow and heat transfer are conducted by using commercial CFD software FLUENT14.0. The results show that the fluid is fully developed after the first trefoil-hole baffle. The heat transfer coefficient and pressure drop vary periodically along the axial direction. Fluid velocity increases gradually and the jet flow forms in the region near baffles. The secondary flow is also produced on the two sides of baffles when the fluid flows through trefoil-hole baffle. The jet flow and secondary flow can decrease the thickness of boundary layer and then enhance the heat transfer.

© 2015 Published by Elsevier Ltd. This is an open access article under the CC BY-NC-ND license

[\(http://creativecommons.org/licenses/by-nc-nd/4.0/\)](http://creativecommons.org/licenses/by-nc-nd/4.0/).

Peer-review under responsibility of Applied Energy Innovation Institute

Keywords: trefoil-hole baffle; shell-and-tube heat exchanger; shell side; heat transfer enhancement; numerical investigation

1. Introduction

Shell-and-tube heat exchangers(STHEs) are widely used in various industrial fields such as petroleum refining, power generation and chemical process, especially nuclear power system[1]. Lots of energy could be saved by improving the structure of these equipment. The segmental-baffle shell-and-tube heat exchangers have been well developed and widely used[2-3]. However, the pressure loss and the ‘dead flow’ region of them are considerably large. Moreover, the harmful vibration and the energy consumption

* Corresponding author. Tel.: +86 21 64253513; fax: +86 21 64253513.E-mail address: zhougy@ecust.edu.cn.

are high. As a new type of longitudinal flow heat exchanger[4], shell-and-tube heat exchangers with trefoil-hole baffles is developed to solve these problems.

In order to understand the underlying mechanism of heat transfer enhancement, Dong [5] proposed the ‘unit channel model’ and numerically investigate the flow and heat transfer characteristics of shell-and-tube heat exchangers with trefoil-hole baffles. The results showed that the heat transfer rate and pressure loss decrease with the increase of the hole height and baffle space. You[6-7] obtained the heat transfer rate and pressure drop formulation on the shell side under specified baffle space. However, these studies mainly focus on the simplification of the geometry and usage of the ‘unit channel’ [8-9], in which the fluid flows in different channels without interference is assumed. So, the heat transfer performance of the whole heat exchangers and the mechanism of the heat transfer enhancement can not be understood and mastered.

To understand the fundamental mechanisms of fluid flow and heat transfer on shellside of shell-and-tube heat exchanger with trefoil-hole baffles, a whole model for numerical simulation is proposed to study the forced convection heat transfer characteristics and pressure drop in the present paper. Compared with the experimental results, the feasibility and accuracy of the proposed model are verified. Furthermore, the flow field and heat transfer enhancement mechanism are also analysed.

2. Geometrical model

Fig. 1 depicts the sketch of a shell-and-tube heat exchanger with trefoil-hole baffles. Trefoil holes are broached on the baffles for the support of tubes. The fluid flows through the gaps between the tubes and baffles longitudinally. The detailed structural parameters are listed in Table 1.

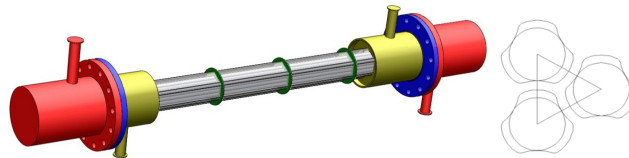


Fig. 1. The geometry of shell-and-tube heat exchanger with trefoil-hole baffles: a) Structure, b) Tube layout

Table 1. Geometric parameters of heat exchanger

Parameter	Value
Inner/Outer diameter of tube/mm	12.2/14
Central distance of tubes/mm	19
Arrangement of tubes	rotational regular triangle
Tube number	30
Tube length /mm	1830
Trefoil-hole baffle number	4
Trefoil-baffle pitch /mm	400
Thickness of trefoil-baffle /mm	5
Diameter of inlet and outlet nozzles /mm	45
Diameter of hexagon inscribed circle /mm	64

3. Numerical model

3.1. The Governing equations

Water is selected as the shellside working fluid. It is treated as an incompressible fluid. The physical properties vary with temperature and are calculated by using the piecewise-linear[10] method to improve computing precision. The flow is assumed to be stable and turbulent. The viscous heating and pressure power are neglected due to the limited fluid velocity and incompressible medium assumption. Based on the above assumptions, the governing equations of fluid flowing on the shellside are presented as following[11-12]:

Continuity equation:

$$\frac{\partial}{\partial x_i}(\rho u_i) = 0 \quad (1)$$

Momentum equation:

$$\frac{\partial}{\partial x_i}(\rho u_i u_k) = \frac{\partial}{\partial x_i}(\mu \frac{\partial u}{\partial x_i}) - \frac{\partial p}{\partial x_k} \quad (2)$$

Energy equation:

$$\frac{\partial}{\partial x_i}(\rho u_i T) = \frac{\partial}{\partial x_i}(\frac{k}{C_p} \frac{\partial T}{\partial x_i}) \quad (3)$$

where T and p stand are fluid temperature and pressure, respectively; u is the fluid velocity; ρ and C_p are fluid density and constant pressure specific heat, respectively; k is heat conductivity coefficient.

Considering the highly swirling on the shellside with non-isotropic turbulence, RNG k - ε turbulence model together with standard wall function[13] is adopted for the current computation[10]. The conservation equations of turbulence kinetic energy and its dissipation rate are given below:

Turbulence kinetic energy k :

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}(\alpha_k \mu_{eff} \frac{\partial k}{\partial x_j}) + G_k + \rho \varepsilon \quad (4)$$

Turbulence kinetic energy dissipation rate ε :

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}(\alpha_\varepsilon \mu_{eff} \frac{\partial \varepsilon}{\partial x_j}) + C_{1\varepsilon}^* \frac{\varepsilon}{k} G_k - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \quad (5)$$

$$\text{where } \mu_{eff} = \mu + \mu_t, \mu_t = \rho c_\mu \frac{k^2}{\varepsilon}, C_{1\varepsilon}^* = C_{1\varepsilon} - \frac{\eta(1-\eta/\eta_0)}{1+\beta\eta^3}, \eta = (2E_{ij} * E_{ij})^{1/2} \frac{k}{\varepsilon}, E_{ij} = \frac{1}{2} \left[\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right].$$

The constants[10]: $C_\mu=0.085$, $C_{1\varepsilon}=1.42$, $C_{2\varepsilon}=1.68$, $\beta=0.012$, $\eta_0=4.38$, $a_k=a_\varepsilon=1.39$.

3.2. The computation domain and boundary conditions

According to the symmetry of the structure, a half of the whole volume is taken as the computation domain, which involved in the shell, together with the inlet and outlet headers and tubes. Since the clearances between tubes and baffles are very small, the leakage effects are expected to be negligible in the computation.

Boundary conditions are set as follows: (1) At the inlet of the shell side, a velocity profile of fully-developed turbulent flow is given, while the fluid temperature is set as uniform, $T_{in}=238\text{k}$. The turbulence intensity is set as $I=3\%$. (2) At the outlet of the shell side, an outflow condition is imposed. (3) No slip and no penetration are specified for all walls. (4) The baffles and the shell wall are adiabatic, the temperature of tube wall is set constant, $T_w=307\text{K}$.

3.3. The mesh and the computation scheme

The 3D grid system is generated by using the commercial software ANSYS. The computation domain, except near-baffles region, is discretized with structured tetrahedral elements. The mesh of the regions adjacent to the tubes and supported plate are refined to meet the requirement, as shown in Fig. 2.

The finite volume method is adopt for the discretization, and all the variables are treated with the second-order upwind scheme[14], except the pressure term with standard scheme[13]. Numerical computations are conducted with the pressure-based solver, and pressure and velocity are coupled with the ‘SIMPLE’ algorithm. The convergent criterions are set as: relative residual of 1E-8 for energy and 1E-5 for other variables.

In order to ensure the accuracy of the calculation, grid independency of the results has been checked. It is found that the relative discrepancies of both Nusselt number Nu and the pressure loss for the latter two are within 1%, which demonstrates that the CFD simulation is of reasonable precision. Thus the grid system of 1.36×10^7 is adopted for the final computation.

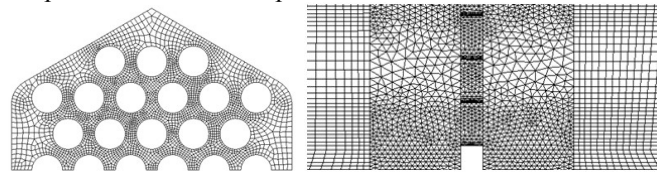


Fig. 2. Meshes of the studied model: a) Grids of cross section, b) Grids near trefoil-hole baffle

4. Model verification

In order to verify the accuracy of the proposed CFD model, an experiment for shell-and-tube heat exchanger with trefoil-hole baffle is carried out.

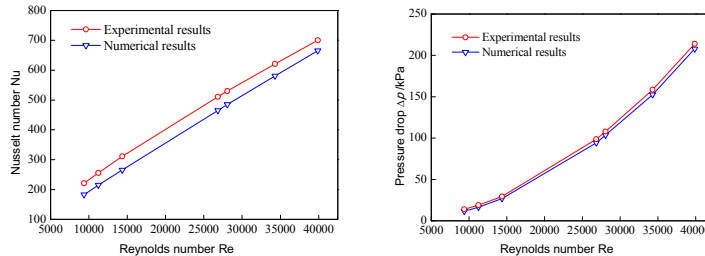


Fig. 3. Comparison of the experimental and numerical results

The experimental results are shown in Fig. 3. It shows that the numerical Nusselt number and the pressure loss are in good agreement with experimental results. The maximum error of Nusselt number and the pressure loss are 17.4% and 20.3%, respectively. It indicated that the proposed physical model and the numerical calculation method is feasible and accurate.

5. Results and discussions

5.1. Heat transfer and pressure drop performance

The mean heat transfer coefficient of the cross section h_{av} is defined as: $h_{av} = q / (T_w - T_{av})$. where q is the average heat flux, T_{av} is the average fluid temperature, T_w is the temperature of tubes.

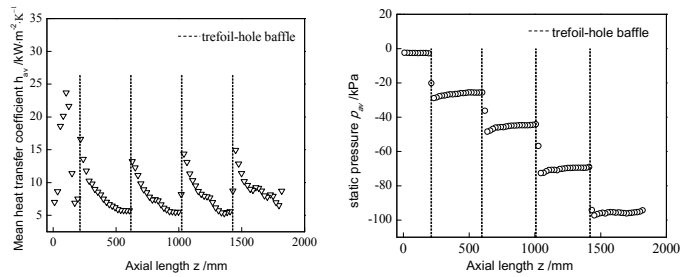


Fig. 4. The variation of a) mean heat transfer coefficient and b) static pressure along with axial direction z

The variation of the average heat transfer coefficient h_{av} , static pressure p_{av} of cross sections along with axial length is shown in Fig. 4. It can be seen that the fluid is fully developed after the first trefoil-hole baffle. The flow and heat transfer characteristics of the central shellside are important and can be used as design rules. In addition, the pressure loss comes from flow resistances and local loss flowing through baffles.

5.2. The flow distribution

Fig. 5 shows the velocity distribution on the longitudinal section. It can be seen that the most of fluid flows longitudinally and uniformly. The acceleration and expansion of the working fluid are clear when it flows cross baffles, and the flow direction varies, which generates multidirectional jet and results in the turbulence enhancement. This is because that the flow area decreases and the velocity increases sharply. Since the block of the baffle, the fluid in front of the baffles flows from the center to the edge of the shell, and the fluid has passed through the baffles flows back to the center from the edge, which produce the secondary flow and enhance the convection heat transfer on shell side effectively.

Due to the effect of shell wall and baffles, the velocity near the inner shell wall is low, and the velocity near the center is high. Along the radial direction, the velocity decreases gradually, which leading to the decrease of the average convective heat transfer coefficient.

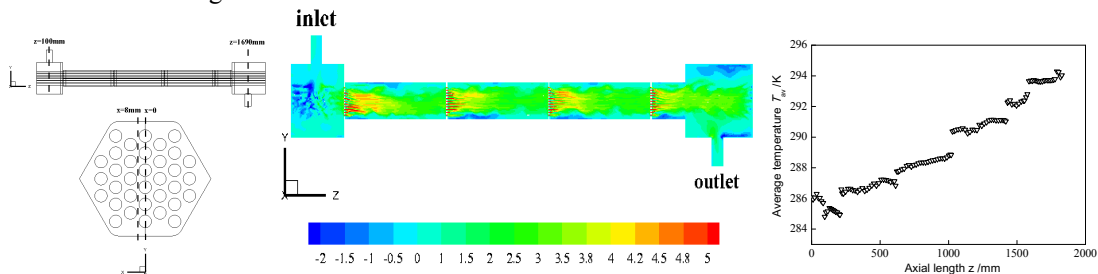


Fig. 5. Flow distribution of the longitudinal section($x=8\text{mm}$) Fig. 6. The variation of average temperature along the axial length

The temperature distribution T_{av} along the axial length is shown in Fig. 6. It can be seen that the temperature near baffles increases obviously due to the generation of the jet and secondary flow. Thus, the local heat transfer is enhanced relatively.

6. Conclusions

The numerical analysis of heat transfer and fluid flows in the shell sides of the shell-and-tube heat exchanger with trefoil-hole baffles is carried out, with the aim to improve the overall thermo-hydraulic performance in longitudinal flow heat exchangers. The following conclusions be drawn from the study:

1) It has been shown that the numerical Nusselt number and the pressure loss are in good agreement with those from experiments. The relative deviation is lower than 17.4% and 20.3%, respectively. It indicates that it is reasonable to use the proposed CFD model and computation method in the analysis of shell-and-tube heat exchanger with trefoil-hole baffles.

2) Due to the structure characteristic, the fluid on the shellside of shell-and-tube heat exchanger with trefoil-hole baffles flows periodically. The gradient temperature and pressure also recur periodically. The fluid flow is fully developed after the first trefoil-hole baffle.

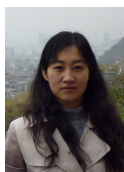
3) For fluid flowing through the baffles, the acceleration and expansion of the working fluid are clear and the jet and swirl flows are generated due to the decrease of the flow area. Along the radial direction, the velocity decreases gradually, which leading to the decrease of the average convective heat transfer coefficient.

Acknowledgements

This study is sponsored by Shanghai Pujiang Program.

References

- [1] Bell K. Heat exchanger design for the process industries[J]. *Journal of Heat Transfer*, 2004, 126(6): 877-885.
- [2] Shi Jingui. Research on flow-field and temperature numerical simulation to shell and tube heat exchanger[D]. Nanjing: Nanjing University of Technology, 2008.
- [3] Muhammad Mahmood Aslam Bhutta, Muhammad Hassan Bashir, Ahmer Rais Khan, Kanwar Naveed Ahmad, Sarfaraz Khan. CFD applications in various heat exchangers design: A review[J]. *Applied Thermal Engineering*, 2012, (32): 1-12.
- [4] Zhang Jianfei, He Yaling, Tao Wenquan. 3D numerical simulation on shell-and-tube heat exchangers with middle-overlapped helical baffles and continuous baffles-Part I: Numerical model and results of whole heat exchanger with middle-overlapped helical baffles[J]. *International Journal of Heat and Mass Transfer*, 2009, 52: 5371-5380.
- [5] Dong Qiwu, Bai Caipeng, Liu Minshan, Wang Dan. Study on flow and heat transfer of shell side of heat exchangers with trefoil baffle. Heat and mass transfer at the Chinese academy of engineering thermal physics learning academic conference proceedings[C]. Xi'an: Chinese Society of Engineering Thermophysics, 2011: 1-5.
- [6] You Yonghua, Fan Aiwu, Lai Xuejiang, Huang Suyi, Liu Wei. Experimental and numerical investigations of shell-side thermo-hydraulic performances for shell-and-tube heat exchanger with trefoil-hole baffles[J]. *Applied Thermal Engineering*, 2013,50: 950-956.
- [7] You Yonghua, Fan Aiwu, Chen Chen, Fang Shuili, Jin Shiping, Huang Suyi. Numerical study of shellside performance of heat transfer and flow resistance for heat exchanger with trefoil-hole baffles[J]. *Advanced Materials Research*, 2012, 557: 2141-2146.
- [8] Dong Q W, Wang Y Q, Liu M S. Numerical and experimental investigation of shellside characteristics for RODbaffle heat exchanger[J]. *Applied Thermal Engineering*, 2008,28 (7): 651-660.
- [9] Dong Q W, Liu M S, Zhao Z D. Research on the characteristic of shellside support structures of heat exchanger with longitudinal flow of shellside fluid[J]. *ASME Trans.*, 2005, 8(2): 1491-1498.
- [10] FLUENT14.0 User's Guide.
- [11] Xiong Q, Li B, Xu J. GPU-accelerated adaptive particle splitting and merging in SPH. *Comput. Phys. Commun.* 2013, (184): 1701-1707.
- [12] Xiong Q, Deng L, Wang W, et al., SPH method for two-fluid modeling of particle-fluid fluidization. *Chem. Eng. Sci.* 2011, 66: 1859-1865.
- [13] Tao Wenquan. Numerical Heat Transfer [M]. 2nd ed. Xi'an: Xi'an Jiao Tong University Press, 2001.
- [14] Barth T J, Jespersen D C. The design and application of upwind schemes on unstructured meshes. *AIAA Paper*, 1989, 89(366): 1-12.



Biography

Dr. Guo-yan Zhou is a Professor of Mechanical Engineering and now working at East China University of Science and Technology. She received her Ph. D. degree from the same university in 2007. Her interests of research include development and optimization of compact heat exchanger, advanced manufacturing technology and MCMS, etc.